Air Distribution in Rooms

research and design methods

Nielsen, Peter V.

Publication date:
1994

Document Version
Publisher's PDF, also known as Version of record

Link to publication from Aalborg University

Citation for published version (APA):
Nielsen, P. V. (1994). *Air Distribution in Rooms: research and design methods*. Dept. of Building Technology and Structural Engineering. Indoor Environmental Technology Vol. R9416 No. 41

INSTITUTTET FOR BYGNINGSTEKNIK
DEPT. OF BUILDING TECHNOLOGY AND STRUCTURAL ENGINEERING
AALBORG UNIVERSITET • AUC • AALBORG • DANMARK

INDOOR ENVIRONMENTAL TECHNOLOGY
PAPER NO. 41

Opening Lecture at ROOMVENT '94, Fourth International Conference on Air Distribution in Rooms, June 15-17, 1994, Cracow, Poland

P. V. NIELSEN

AIR DISTRIBUTION IN ROOMS - RESEARCH AND DESIGN METHODS

JULY 1994
ISSN 0902-7513 R9416

INSTITUTTET FOR BYGNINGSTEKNIK
DEPT. OF BUILDING TECHNOLOGY AND STRUCTURAL ENGINEERING
AALBORG UNIVERSITET • AUC • AALBORG • DANMARK

INDOOR ENVIRONMENTAL TECHNOLOGY
PAPER NO. 41

Opening Lecture at ROOMVENT '94, Fourth International Conference on Air Distribution in Rooms, June 15-17, 1994, Cracow, Poland

P. V. NIELSEN

AIR DISTRIBUTION IN ROOMS - RESEARCH AND DESIGN METHODS

JULY 1994 ISSN 0902-7513 R9416

AIR DISTRIBUTION IN ROOMS - RESEARCH AND DESIGN METHODS

Peter V. Nielsen
Department of Building Technology
and Structural Engineering
Aalborg University
Aalborg, Denmark

SUMMARY

The research on air distribution in rooms is often done as full-size investigations, scale -model investigations or by Computational Fluid Dynamics (CFD). New activities have taken place within all three areas and this paper draws comparisons between the different methods. The outcome of the IEA sponsored research "Air Flow Pattern within Buildings" is used for comparisons in some parts of the paper because various types of experiments and many countries are involved.

The trend in design methods is illustrated by full-size experiments, model experiments, CFD-methods and flow element methods. The CFD-method has grown in popularity for large projects but the flow element method is still developing for the more general situations.

AIR DISTRIBUTION IN ROOMS - RESEARCH AND DESIGN METHODS

Peter V. Nielsen
Department of Building Technology
and Structural Engineering
Aalborg University
Aalborg, Denmark

INTRODUCTION

This paper will show some aspects of the present state of research on air distribution in rooms and it will also show design methods which are used in practice.

The research on air distribution in rooms can be divided into the areas: full-scale experiments, scale-model experiments and Computational Fluid Dynamics, (CFD). Some experiments are carried out to get an understanding of the whole air movement in a ventilated room and others (physical or computer based experiments) are carried out to get an understanding of the process in an isolated flow area as e.g. a jet, a plume or a stratified layer. Design methods can either be based on full-scale or model experiments or on CFD, but they can also be based on results from the work on isolated flow areas called "flow elements" as shown in the end of this paper.

Many activities are taking place all over the world in the field of air distribution experiments and computer simulation and it is therefore difficult to make a selection for this presentation. One of the important events is the activities sponsored by the International Energy Agency (IEA) during the period 1988 and 1991 under the name "Air Flow Pattern within Buildings". A number of countries joined a research cooperation which performed both full-scale experiments, scale-model experiments and Computational Fluid Dynamics in a single geometry in a few different cases. The presentation is therefore based on this work, but other typical results and methods are described as examples to give a broad idea of the whole field.

FULL-SCALE EXPERIMENTS

Full-scale experiments will produce measurements which can be of very high quality because they are close to the real situation in buildings. The experiments are time-consuming and they involve high investments because it is necessary to use expensive equipment.

The low velocity level makes measurement of the velocity difficult and it is necessary to use equipment as hot-wires or hot-spheres. This equipment can only be used in areas where the turbulence is restricted to $\sim$ 30% in comparison with the air velocity. It is therefore necessary to use smoke tests together with the velocity measurements to get a broad idea of the air movement in the room and to identify areas with sufficient velocity in comparison with turbulence. Those areas are typical free jets and wall jets, plumes, cold downdraught and recirculating flow close to walls and floor. The centre part of a ventilated room will often be an area with low velocity and high turbulence.

Laser-doppler anemometry is a new possibility for measurement of velocity and turbulence which especially can be used everywhere in the room at any velocity and turbulence level.

The visualization of the air movement can be made by smoke which at any location can be supplied through a pipe. The smoke will indicate the streaklines of the flow and it will show the movement of any contaminant from a source in the room. It can be illuminated by a light slit or by a laser slit, in the latter case with only a little heat absorption on the surfaces. There are also examples where the visualization is made by helium bubbles or by metaldehyde particles.

Temperature measurements are made by thermocouples, resistance measurements or by cold-wires. The method in the latter case is used if it is necessary to measure the fluctuating component of the temperature. It is important in special cases to be aware of the level of mean radiant temperature. Surface temperature distribution can be found in a quick way by the registration of infrared radiation (thermovision).

Tracer gas measurements are used for the study of contaminant transport and for the measurements of ventilation effectiveness and air exchange efficiency. They can be made continuously where tracer gas is released in points and the air is analysed in different measuring points. It is also possible to use a passive tracer gas technique which involves an analysis of samples by gaschromatography after the experiment. The geometrical details around the source, emission rate, tracer gas density are all very important in an experiment with contaminant transport.

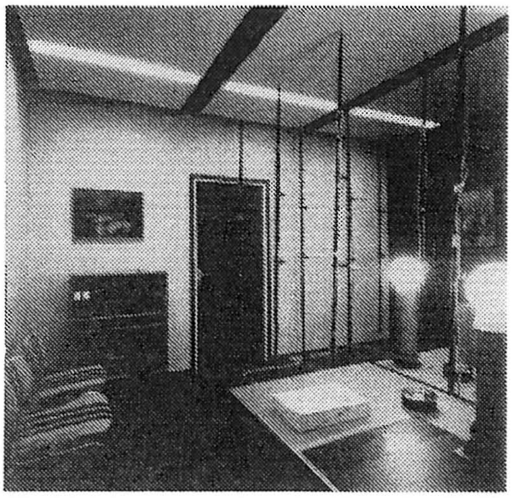

Figure 1. Full-scale experiments with an office room for the Ministry for Foreign Affairs in Denmark. The experiments are carried out by ABB Fläkt.

Figure 1 shows the set-up in a full-scale experiment of the more expensive type. Experiments will of course involve the registration of thermal comfort in different situations, but they may also involve a test of actual decoration, furnishings and interior fittings, so the owner, architect and consultants can experience the environment at a very early stage as discussed by Lärkfeldt [1]. This is of course an extra advantage of full-scale experiments which also will ensure an accurate determination of the flow in the occupied zone and give information about the influence from different obstacles in the interior of the room such as lamps, technical installations, etc.

It may obviously be difficult to perform full-scale experiments in very large rooms due to necessary space requirements in the laboratory. Some full-scale experiments can be performed as small parts of the entire flow field. If we are concerned about problems as e.g. draught from a cold air jet which flows directly into the occupied zone it is only necessary to study the part of the space which is involved in the local problem and this may for instance be a cold window, obstacles, tables, etc.

In the following some of the measurements made in the IEA project "Air Flow Pattern within Buildings" will be shown. The experiments are made in full-scale test rooms and the emphasis is put on the air movement measurements without the detailed furnishings shown in figure 1. The measurements are made in four different countries in rooms with the dimensions H, L and W equal to 2.5, 4.2 and 3.6 m. (The Danish test room has the height: 2.4 m). The supply opening in the experiments is a wall-mounted device which consists of 84 small nozzles. The supply opening will give a complicated flow close to the nozzles which is typical of an advanced air terminal device and this is important for the validation of the full-scale experiments, scale-model experiments and CFD-predictions.

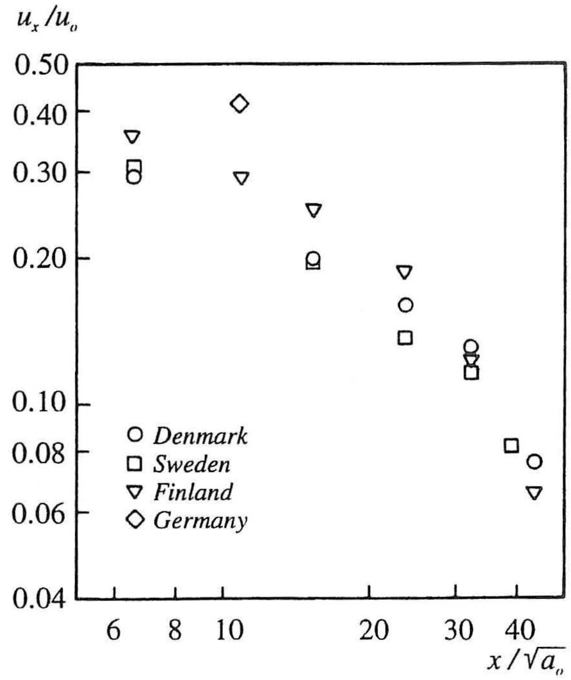

Figure 2. Velocity decay versus distance in the wall jet below the ceiling measured in four different countries. Isothermal flow, references [2,3].

The measurements in figure 2 show the velocity decay in the centre line of the wall jet below the ceiling. There are some deviations in the measured values, but it should be considered that the deviations contain the effect of different rooms, different measuring equipment, individual diffusers and individual installations.

The maximum velocity in the occupied zone u_{rm} is an important design parameter. Figure 3 shows the level of this velocity as a function of the air change rate measured in three countries. It is possible to see that u_{rm} is proportional to the flow rate at high velocities which indicates fully turbulent flow, while it has a low turbulent effect for a velocity level below 0.15 m/s.

The IEA research work includes full-size measurements of concentration distribution. Heiselberg and Bergsøe [4] show measurements with both conventional tracer gas and passive tracer gas. The measurements show the importance of the boundary conditions and they also show a low turbulence effect in the concentration distribution at least up to $n = 3h^{-1}$.

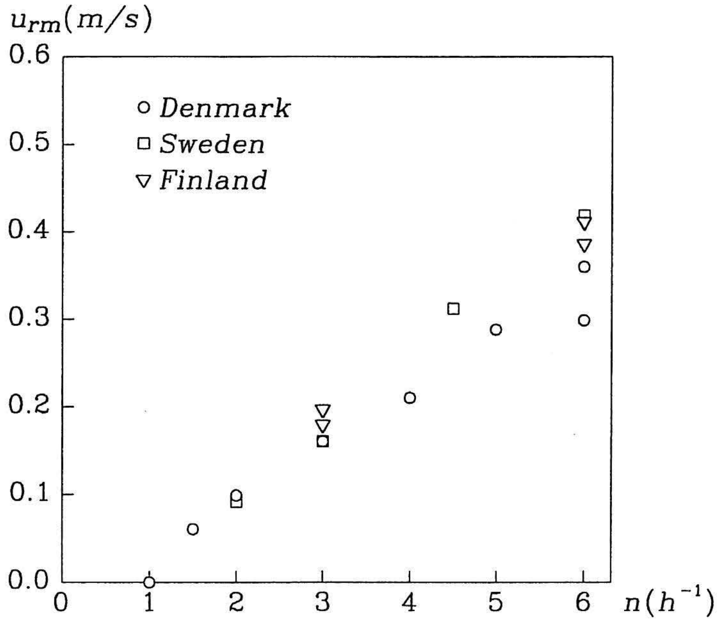

Figure 3. Maximum velocity in the occupied zone u_{rm} versus air change rate in the case of isothermal flow. References [2,3].

SCALE-MODEL EXPERIMENTS

Scale-model experiments can be used for the determination of air flow in large spaces as e.g. shopping arcades, atria and exhibition buildings. It may also be a useful method when a large number of designs have to be tested for the general flow pattern.

It can be shown from theory that a similar flow will be obtained in full scale and in model if the dimensionless boundary conditions are identical and if the Archimedes number, the Reynolds number and the Prandtl number are the same in both situations.

It is impossible to make a model experiment in a strongly reduced scale if all the dimensionless numbers have to be kept constant. If e.g. the scale is reduced by a factor of 10, then the velocity has to be increased by a factor of 10 due to the Reynolds number which will give an increase in the temperature difference by a factor of 1000 in order to keep the Archimedes number. The Prandtl number is, on the other hand, unchanged when air is used as the fluid in the model experiments.

The problem can in practice be overcome if the Reynolds number is high and the flow pattern is governed mainly by fully developed turbulence. It is possible to ignore the Reynolds number and the Prandtl number because the structure of the turbulence and the flow pattern at a sufficiently high level of velocity will be similar at different supply velocities and therefore independent of the Reynolds number. The transport of thermal energy by turbulent eddies will also dominate the molecular diffusion and will therefore be independent of the Prandtl number. See e.g. reference [5] for a further discussion of scale-model experiments.

Thermal radiation and conduction through surfaces are special problems because it is difficult to formulate conditions for model experiments which include those parameters. It is also difficult to make a correct model of an air terminal device as discussed by Nielsen [5].

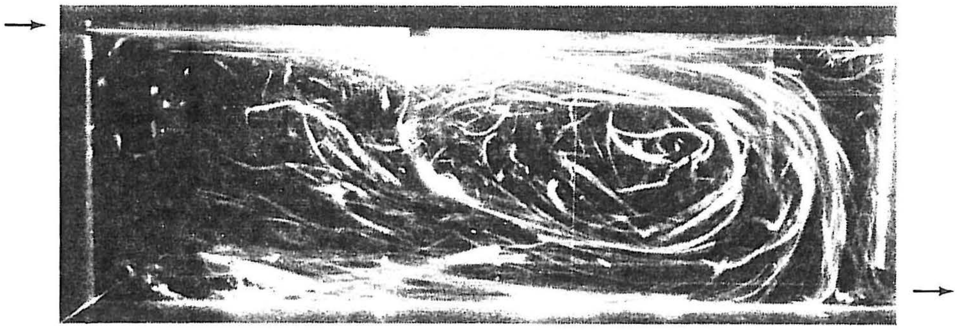

Figure 4. Recirculating flow in a scale-model with an obstacle on the ceiling. The air is supplied through a slot on the left side and the flow is two-dimensional and isothermal.

Figure 4 shows the recirculation flow in a room with an obstacle. A full understanding of the plane isothermal flow in a room with an obstacle will require a large number of experiments and the figure shows one single situation. Model experiments will be useful in this situation because it is easy to change the geometry and reference [6] shows the outcome of measurements of about 100 different sorts of geometries.

Scale-model experiments are important for the research and design of air movement in large spaces. Figure 5 shows a single example where the flow is studied in the restaurant and the exhibition hall of the Danish Pavilion in Seville. The building has two main elements, a steel framed structure, facing west, with a floor area of 45 m × 2.5 m and a height of 24 m and a fibre glass "sail" construction, facing east, which leans against the steel structure. The large room formed between those two structures is enclosed by glass to the north and to the south and it forms the restaurant and the exhibition hall. The space is ventilated by an inlet wall with cooling elements, see figure 5 B and an extract fan in the top north end of the room. Model experiments show a reasonable temperature and velocity level in the space, see reference [7].

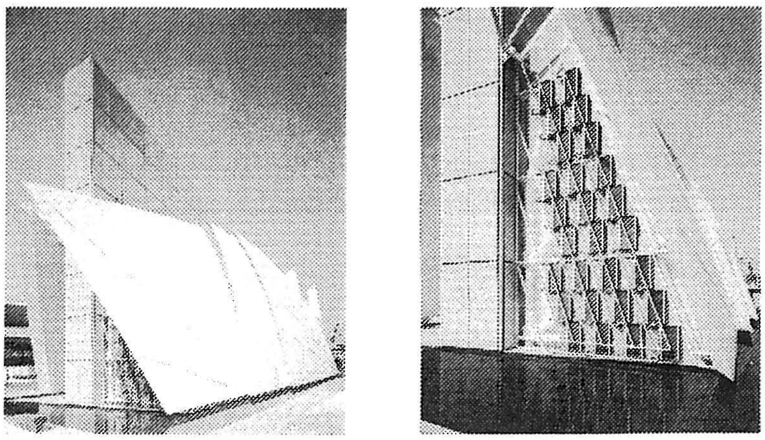

Figure 5. A: The Danish Pavilion at the World Exposition EXPO'92 in Seville. B: Cooling elements and supply openings in the south gable.

France joined the IEA project with model experiments performed by Fontaine et al. [8], in a water model. The measured velocity decay u_x/u_o = 0.22 for $x/\sqrt{a_o}$ = 22.7 can be compared with the full-size measurements in figure 2 and the measured velocity u_{rm} = 0.14 m/s for $n = 3h^{-1}$ can be compared with the full-scale experiments in figure 3.

It is seen from the figures that the French model experiments are in good agreement with full-size experiments from the other countries in the IEA project, especially in the determination of u_{rm}.

The measuring equipment for model experiments is of the same type as the equipment used for full-size investigations. It is typical that the velocity level in model experiments is high compared with full-size experiments because the Reynolds number must have the same size in the two situations or at least a certain size. A high velocity level makes a high degree of accuracy, but the problems with the high turbulent level remain. An increased "real time" in model experiments with air makes it difficult to follow smoke movements, but the use of particles may be another possibility as shown in figure 4.

COMPUTATIONAL FLUID DYNAMICS

The fluid dynamics research is strongly influenced by the increasing computer power which has been available for the last decades. This development has the effect that the cost for a given job will decrease by a factor 10 every eight years. The development shows not only a decreasing cost but the computer time is also decreasing. Chapman cites an impressive example of this trend in computing efficiency, [9]. He mentioned that a numerical calculation of the flow over an airfoil would take 30 years if it was started in 1960 and it would cost $ 10 million. Twenty years later - in 1980 - the same calculation would take half an hour and cost $ 1000. The same calculation will not be worth mentioning today in the 'nineties.

There are various reasons for this development. Firstly, the computer speed is increasing more rapidly than the computer costs and this tendency seems to continue. Secondly, a process takes place which increases the flexibility of different software as pre- and post-processor software and, furthermore, there is a continuous development of new software. Improvements in the fundamental routines as e.g. the grid generation procedure and the numerical method do also contribute to an increasing speed.

The above-mentioned tendencies have also influenced the indoor environmental technology. One of the first examples of a prediction based on Computational Fluid Dynamics (CFD) in indoor environmental technology was internationally published by Nielsen in 1973 [10]. The activities have increased dramaticly since that time and especially during the last years. It can be mentioned that all CFD-papers at the first ROOMVENT conference in Stockholm in 1987 were presented within a single session, while half of all papers at the third ROOMVENT conference in Aalborg in 1992 were based on, or included, CFD-calculations.

Eleven countries joined the CFD-part of the IEA project on air flow pattern within buildings. Figure 6 shows the computer prediction of mean velocity u_m in the occupied zone versus the air change rate in the case of isothermal flow found by the countries.

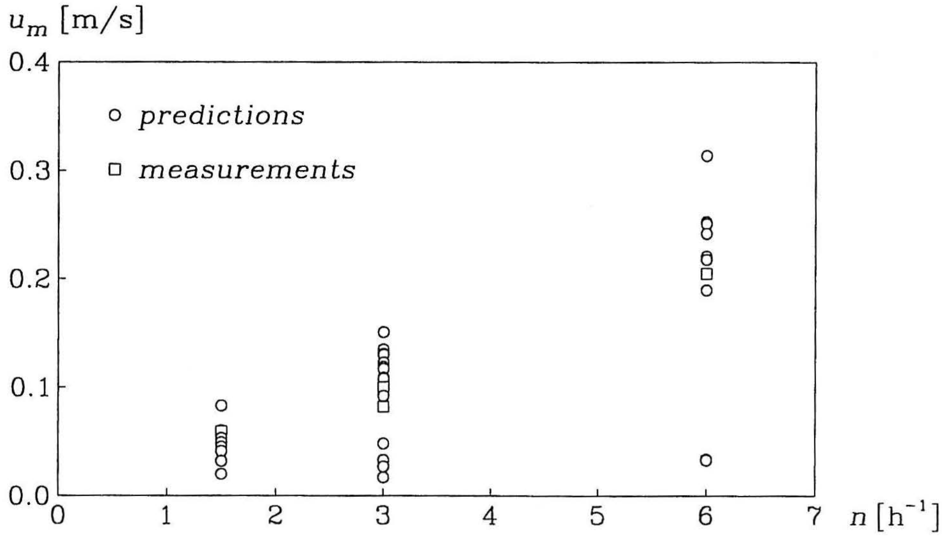

Figure 6. Mean velocity u_m versus air change rate n for the IEA project. Isothermal flow, [2,11].

It is seen from the figure that there is a large variation in the results from the different countries. The variation may be a result of different codes, different descriptions of boundary conditions and perhaps also a result of different grid distributions and number of grid points. The description of boundary conditions at the diffuser may be one of the most important problems because the flow close to the diffuser is very complicated to describe, although it is a typical situation for a practical air terminal device.

Improvements of the boundary condition at the supply opening will be one of the remedies for better predictions. Figure 7 shows the results for a Simplified Boundary Condition (SBC) where the supply area, momentum flow and direction of speed are preserved in the boundary conditions. Comparison with measurements shows that the maximum velocity in the occupied zone is overestimated by 40%.

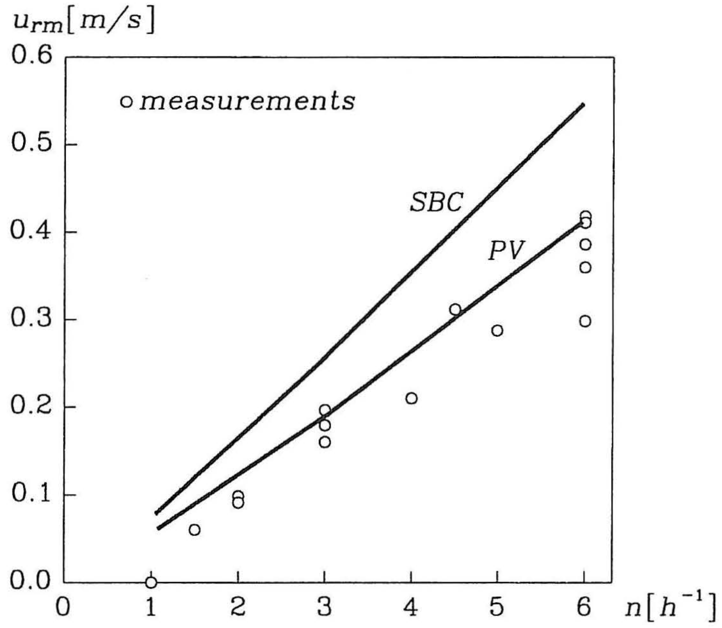

Figure 7. Maximum velocity in the occupied zone versus air change rate. Measurements and predictions in connection with the IEA, Annex 20 work. SBC: Simplified boundary conditions. PV: Prescribed velocity method. Reference [12].

Figure 7 shows that it is possible to make a large improvement of the predictions by use of a prescribed velocity method (PV), see reference [13]. The boundary conditions are given by an analytic description of one or two of the velocity components in a small volume in front of the opening. All other variables are predicted by the numerical method.

Continuous development of the computer capacity will undoubtedly make the direct methods as e.g. local grid refinement possible in the future.

Numerical prediction is able to handle furnished rooms. Figure 8 shows two sets of simulations in a room with air terminal device and dimensions according to the IEA-experiments. The jet below the ceiling is almost uninfluenced by the obstacle but there is a redistribution of the flow in the lower part of the room. The level of the maximum velocity in the occupied zone is not highly influenced by the obstacle. The predictions are in good agreement with measurements, see Brohus [14].

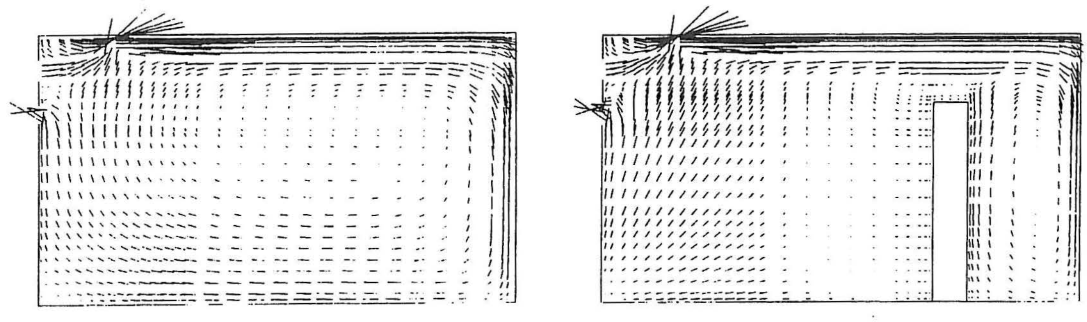

Figure 8. Empty room and room with an obstacle in the occupied zone. Isothermal flow, [14].

The model experiment shown in figure 4 where the recirculating flow is interrupted by a ceiling-mounted obstacle is also a situation which today can be handled by CFD. Figure 9 shows the measurements and predictions of the velocity profile at a location below the ceiling on the downstream side of the obstacle and in the occupied zone in the area with maximum velocity. The predictions seem to be in good agreement with the measurements from 1983 and CFD is a tool which can be used for fast predictions of a variety of geometries in the same way as scale-model experiments were used earlier.

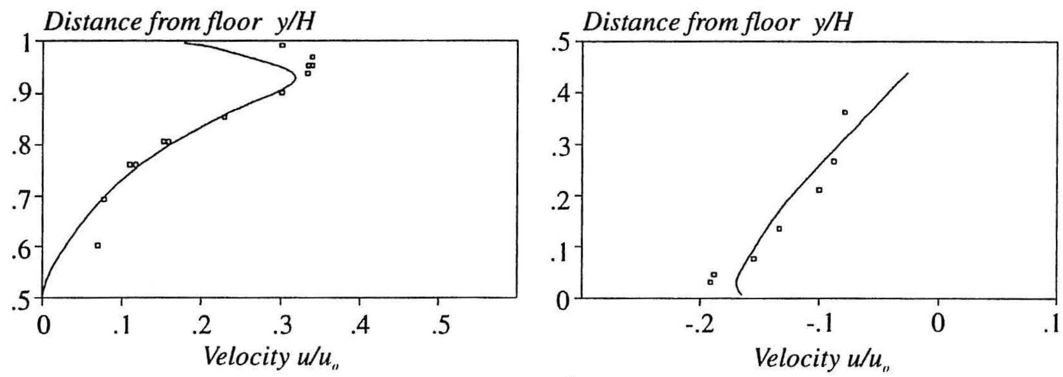

Figure 9. Velocity profile at the ceiling $x/H = 1.7$ and in the occupied zone $x/H = 2.1$. Two-dimensional isothermal flow in a model with the same dimensions as the model in figure 4. Measurements by Nielsen [6] and predictions by Svidt [15].

The work within the IEA-project generally concluded that two-dimensional isothermal flow is well-predicted by CFD and that skill and experience are required to use codes for practical three-dimensional situations with isothermal and non-isothermal flow. It is difficult to deal with natural and mixed convection close to cold or warm surfaces. It is shown that it is not always valid to compute only half a room under symmetrical boundary conditions because the air movement may be unsymmetrical in full-size flow. It should be recognized that two-dimensional flows are rare and that three-dimensional simulations may be needed to investigate characteristics which may be of interest.

CFD is, like scale-model investigations, a possibility when air distribution systems are designed in large enclosures as e.g. shopping arcades and atria. Many examples are shown in the literature as e.g. Tokyo International Forum which is an impressive example given by Murakami [16]. The atrium space is shaped like a big ship with glass walls and it has a length of 200 m and a volume of 200000 m^3.

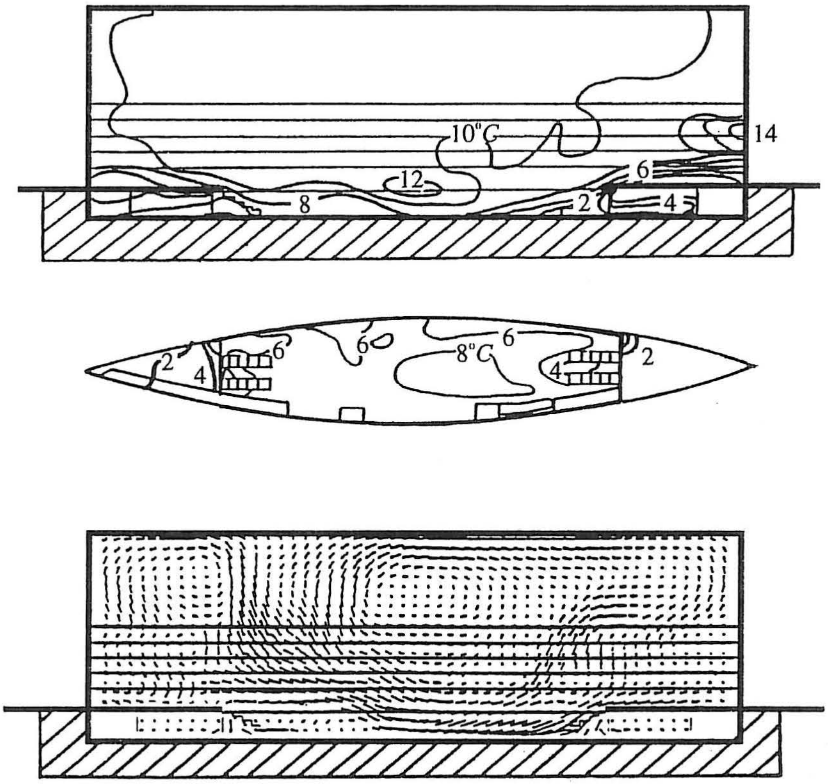

Figure 10. Temperature and air flow distribution under heating conditions. Tokyo International Forum.

The numerical simulation in figure 10 is made by three subsystems. The air flow and temperature distribution are found by CFD based on a $k-\varepsilon$ turbulence model. Radiation heat transfer is simulated between the wall boundaries and the effect of direct solar radiation, diffused solar radiation and diffused reflected radiation has been taken into consideration in the complex shape of the enclosure. The figure shows winter heating conditions with an induced ventilation arising from wind pressure on the entrance.

DESIGN WITH FLOW ELEMENTS

The design of an air distribution system can be based on full-size investigations, scale-model investigations or CFD simulations, but the most convenient method in many practical situations is to design the system from the flow element theory. The flow elements are isolated volumes where the air movement is controlled by a restricted number of parameters and the air movement is fairly independent of the general flow in the room. A free jet from a supply opening is a typical example because the flow is governed by the supply movement flow and it can typically be described by parabolic equations.

A number of flow elements are shown in figure 11. The first group of isothermal flow elements consists of a free jet, A, wall jet, B, wall jet with redistribution of flow close to the opening, C, deflection of a jet by a wall, D, and deflection of a jet by an obstacle, E. Figure 11 F shows the local flow close to a return opening.

The non-isothermal flow elements can be trajectory and velocity in a free jet, G, and penetration length in a cold horizontal wall jet, H.

Another group of flow elements is buoyant flow and gravity induced flow. Figure 11 I indicates the plume above a heat source which is given as a function of convective heat release and vertical temperature gradient. Free convection around a person is a flow element, which can connect the exposure to the concentration distribution, J, and the last example shows the cold downdraught which is a gravity induced flow, K.

New research has shown that stratified flow also can be treated as flow elements. Figure 11 L shows the velocity distribution at the floor from a wall-mounted low-velocity inlet for displacement ventilation and a similar flow is obtained from cold downdraught, M. The return opening in stratified surroundings, N, is another example of a special flow different from the isothermal flow in a similar geometry.

We will use the flow element method to make a dimensioning of the air distribution system in a room identical to the test room in the IEA work because we have the possibility to evaluate the results.

One method is to make a design according to the throw of the isothermal wall jet, see figure 11 B. The throw of an isothermal jet is the length of the jet to a given reference velocity. It is the purpose of the design procedure to control the air distribution in the room in such a way that the maximum velocity in the occupied zone u_{rm} is up to 0.15 m/s. General experience shows that this is achieved when the throw is equal to room length and the reference velocity in the wall jet is equal to 0.2 m/s. A design based on throw, with the supply opening tested in figure 2, will give a maximum velocity in the occupied zone of u_{rm} = 0.08 m/s when the flow is isothermal [3].

A design based on CFD and simplified boundary conditions will give a velocity of 0.09 m/s and the use of prescribed velocity will give a velocity of 0.14 m/s, which is closer to the expected velocity of 0.15 m/s, see [3].

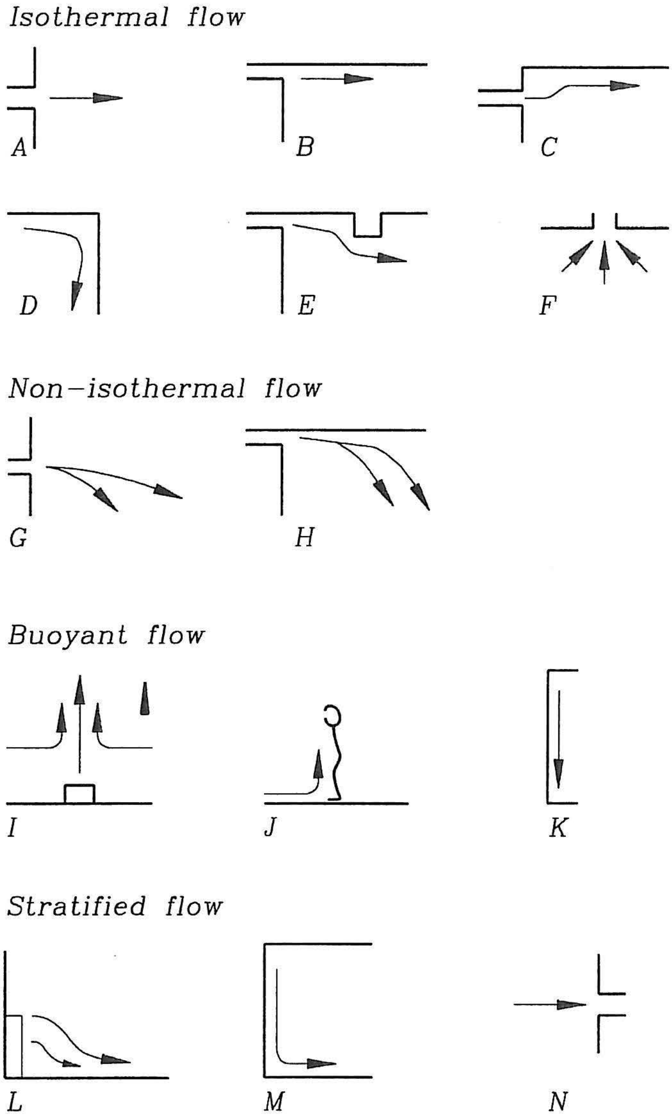

Figure 11. Flow element in a room. The elements are divided into isothermal flow, non-isothermal flow, buoyant flow and stratified flow.

The flow element method can be improved by considering the velocity decay in the deflected jet at the end wall, see figure 11 D. The sketch in the left side of figure 12 shows the flow at the end wall opposite the supply opening in the case of a two-dimensional flow. The deflected jet flows vertically down the end wall as a new two-dimensional flow and u_{rm}/u_L is equal to 0.7 where u_L is the reference velocity in a non-deflected jet with a length of L. The sketch in the right side of figure 12 shows the air movement at the end wall where the flow is three-dimensional. The deflected jet flows down the wall as a semi-radial jet and experiments and numerical predictions show that u_{rm}/u_L varies between 0.3 and 0.7. u_{rm}/u_L is a function of jet width divided by the width of the end wall. Large jet width corresponds to two-dimensional flow ($u_{rm}/u_L \sim 0.7$) and a small width of the primary jet corresponds to radial flow at the end wall ($u_{rm}/u_L \sim 0.3$).

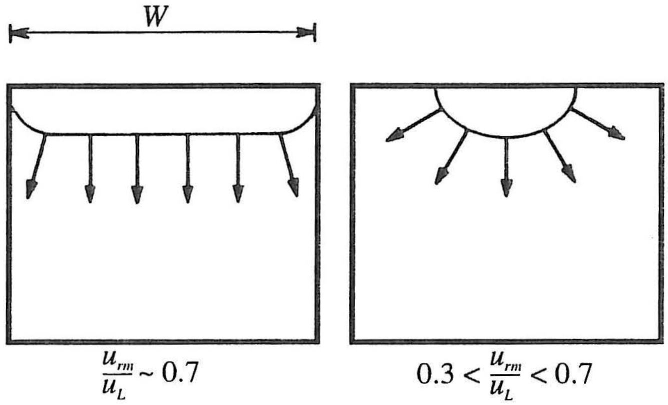

Figure 12. Flow at the end wall opposite the supply opening in the case of two-dimensional and three-dimensional air movement.

A design according to maximum velocity in the occupied zone using figure 13 as a design chart will give an air change rate in the room of $n = 3.8 h^{-1}$ which corresponds to a maximum velocity in the occupied zone of $u_{rm} \sim 0.2$ m/s according to figure 3.

The curve in figure 13 is made by numerical prediction of the flow in rooms of different sizes with a nozzle as a supply opening. The measured result of $u_{rm}/u_L = 0.45$ for the IEA-test case is also shown in figure 13 and the use of this value will give the correct velocity in the occupied zone.

Reference [3] shows the use of the penetration length, figure 11 H, as a design flow element. It is concluded from the IEA-tests that the general behaviour of the cold jet is predictable but further research is needed to use this simplified method as part of a general design procedure.

Design of displacement ventilation involves flow elements as plumes and cold downdraught. New research has shown that it is possible to handle the stratified flow from a wall-mounted low-velocity supply opening as a flow element which is relatively independent of other elements and room dimensions. Research on two- and three-dimensional stratified flow from supply opening and a corresponding flow from cold downdraught are treated by Nielsen [17] and Heiselberg [18].

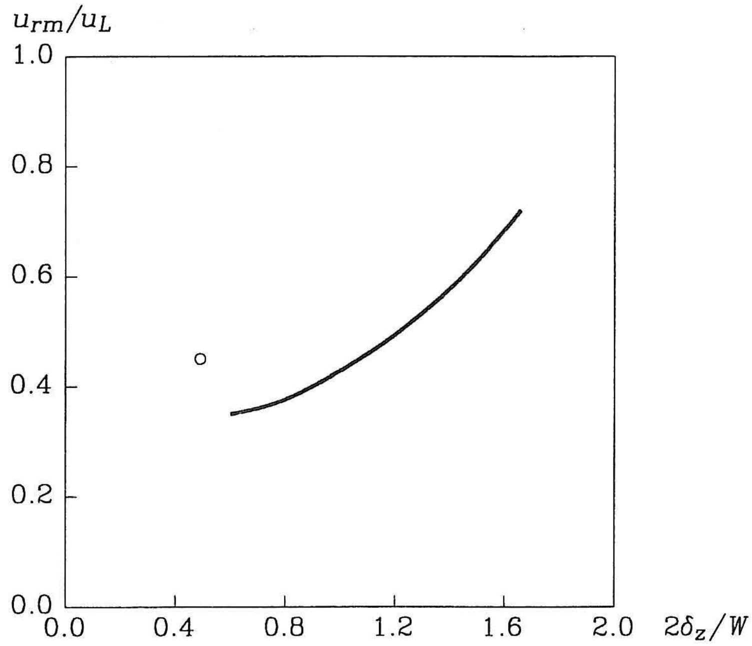

Figure 13. Velocity ratio u_{rm}/u_L for isothermal three-dimensional flow, at the wall opposite the supply wall, versus relative width of jet $2\delta_z/W$.

CONCLUSIONS

Full-size investigations will produce results which can be of a very high quality because the situation can be very close to the real situation in the building. In different countries measurements on a given air terminal device in rooms of identical size show good agreement when they are compared, even in the low velocity region.

Scale-model experiments are useful in cases where a large number of parameters are varied.

The air movement in large enclosures can be investigated in a model but special attention must be paid to radiation, low-turbulent effect and geometrical details around supply openings.

Results from model experiments show agreement with full-size measurements in the IEA research project.

Computational Fluid Dynamics is used increasingly for the prediction of room air distribution. It is shown that skill and experience are required for the use of the codes in the practical three-dimensional situation and it is difficult to deal with supply openings and heat transfer at surfaces.

Two-dimensional flow is well-predicted but it is rare in practical situations.

CFD-prediction of air movement in large enclosures is a very important alternative to model experiments, especially in cases where the area of interest is energy flow and mass flow. The prediction of energy flow and mass flow is less demanding to obtain in comparison with an exact prediction of the velocity level.

Many practical design methods are based on the flow element method. Design according to the throw of an isothermal jet is one of the basic methods which gives a fair result. The method can be improved if it is supplied with information on details of the air movement outside the main jet flow from the air terminal device.

The research into flow elements and the use of the design method based on flow elements are still developing. Stratified flow can be mentioned as one of the new research areas.

REFERENCES

[1] Lärkfeldt, B. "**Full-Scale Tests for Determining Air Movements and Temperature Distributions**". Proc. of the Int. Conference on Air Distribution in Ventilated Spaces, ROOMVENT '87, Stockholm, 1987.

[2] Lemaire, A.D. (Editor). "**Room Air and Contaminant Flow, Evaluation of Computational Methods**". Subtask-1 Summary Report, TNO Building and Construction Research, Delft, 1993.

[3] Nielsen, P.V. "**Models for the Prediction of Room Air Distribution**". Proc. of the 12th AIVC Conference, ISBN 0 946075 53 0, AIVC, Warwick, 1991.

[4] Heiselberg, P. and Bergsøe, N.C. "**Measurements of Contaminant Dispersion in Ventilated Rooms by a Passive Tracer Gas Technique**". Room Air Convection and Ventilation Effectiveness, ASHRAE, 1993.

[5] Nielsen, P.V. "**Model Experiments for the Determination of Airflow in Large Spaces**". Proc. of the Int. Conf. INDOOR AIR '93, Helsinki, 1993.

[6] Nielsen, P.V. "**Air Diffusion in Rooms with Ceiling-Mounted Obstacles and Two-Dimensional Isothermal Flow**". Proc. of the 16th Int. Conf. on Refrigeration, Paris, 1983.

[7] Fox, S.G. and Nielsen P.V. "**Model Experiments in 1990 and On-Site Validation in 1992 of Air Movement in the Danish Pavilion in Seville**". Proc. of the Int. Conf. INDOOR AIR '93, Helsinki, 1993.

[8] Fontaine, J.R., Biolley, F., Rapp, R. and Serieys, J.C. "**Ventilation Flow Analysis-Flow Visualization and LDA Measurements in Water Scale Models, Validation of Numerical Results**". Proc. of the 12th AIVC Conference, ISBN 0 946075 54 9, AIVC, Warwick, 1991.

[9] Chapman, D.R. "**Computational Aerodynamics Development and Outlook**". AIAA J., Vol. 17, 1979.

[10] Nielsen, P.V. "**Berechnung der Luftbewegung in einem zwangsbelüfteten Raum**". Gesundheits-Ingenieur, 94, No. 10, 1973.

[11] Whittle, G.E. "**Evaluation of Measured and Computed Test Case Results from Annex 20, Subtask 1**". Proc. of the 12th AIVC Conference, ISBN 0946075 53 0, AIVC, Warwick, 1991.

[12] Skovgaaard, M. and Nielsen, P.V. "**Modelling Complex Inlet Geometries in CFD - Applied to Air Flow in Ventilated Rooms**". Proc. of 12th AIVC Conf. ISBN 0 946075 55 7, AIVC, Warwick, 1991.

[13] Nielsen, P.V. "**Description of Supply Openings in Numerical Models for Room Air Distribution**". ASHRAE Transaction, Vol. 98, Part 1, 1992.

[14] Brohus, H. "**Air Flow in a Room Ventilated after the Mixing Principle with an Obstacle Placed in the Occupied Zone**". Fifth Nordic Seminar on Computational Mechanics, Aalborg, 1992.

[15] Svidt, K. "**Numerical Prediction of Buoyant Air Flow in Livestock Buildings**". Proc. of the Fourth International Livestock Environment Symposium, Coventry, 1993.

[16] Murakami, S. "**Prediction, Analysis and Design for Indoor Climate in Large Enclosures**". Proc. of the Third International Conference on Air Distribution in Rooms, ROOMVENT '92, Copenhagen, 1992.

[17] Nielsen, P.V. "**Stratified Flow in a Room with Displacement Ventilation and Wall-Mounted Air Terminal Devices**". ASHRAE Transaction, V. 100, Pt. 1, 1994.

[18] Heiselberg, P. "**Stratified Flow in Rooms with a Cold Vertical Wall**". ASHRAE Transaction, V. 100, Pt. 1, 1994.

PAPERS ON INDOOR ENVIRONMENTAL TECHNOLOGY

PAPER NO. 16: C. E. Hyldgaard: *Water Evaporation in Swimming Baths.* ISSN 0902-7513 R9045.

PAPER NO. 17: H. Overby, M. Steen-Thøde: *Calculation of Vertical Temperature Gradients in Heated Rooms.* ISSN 0902-7513 R9046.

PAPER NO. 18: P. V. Nielsen, U. Madsen, D. Tveit: *Experiments on an Exhaust Hood for the Paint Industry.* ISSN 0902-7513 R9146.

PAPER NO. 19: L. Germann Pedersen, P. V. Nielsen: *Exhaust System Reinforced by Jet Flow.* ISSN 0902-7513 R9147.

PAPER NO. 20: P. V. Nielsen: *Models for the Prediction of Room Air Distribution.* ISSN 0902-7513 R9148.

PAPER NO. 21: M. Skovgaard, P. V. Nielsen: *Modelling Complex Inlet Geometries in CFD - Applied to Air Flow in Ventilated Rooms.* ISSN 0902-7513 R9149.

PAPER NO. 22: M. Skovgaard, P. V. Nielsen: *Numerical Investigation of Transitional Flow over a Backward Facing Step using a Low Reynolds Number $k - \varepsilon$ Model.* ISSN 0902-7513 R9150.

PAPER NO. 23: P. Kofoed, P. V. Nielsen: *Auftriebsströmungen verschiedener Wärmequellen - Einfluss der umgebenden Wände auf den geförderten Volumenstrom.* ISSN 0902-7513 R9151.

PAPER NO. 24: P. Heiselberg: *Concentration Distribution in a Ventilated Room under Isothermal Conditions.* ISSN 0902-7513 R9152.

PAPER NO. 25: P. V. Nielsen: *Air Distribution Systems - Room Air Movement and Ventilation Effectiveness.* ISSN 0902-7513 R9250.

PAPER NO. 26: P. V. Nielsen: *Description of Supply Openings in Numerical Models for Room Air Distribution* ISSN 0902-7513 R9251.

PAPER NO. 27: P. V. Nielsen: *Velocity Distribution in the Flow from a Wall-mounted Diffuser in Rooms with Displacement Ventilation.* ISSN 0902-7513 R9252.

PAPER NO. 28: T. V. Jacobsen & P. V. Nielsen: *Velocity and Temperature Distribution in Flow from an Inlet Device in Rooms with Displacement Ventilation.* ISSN 0902-7513 R9253.

PAPER NO. 29: P. Heiselberg: *Dispersion of Contaminants in Indoor Climate.* ISSN 0902-7513 R9254.

PAPER NO. 30: P. Heiselberg & N. C. Bergsøe: *Measurements of Contaminant Dispersion in Ventilated Rooms by a Passive Tracer Gas Technique.* ISSN 0902-7513 R9255.

PAPER NO. 31: K. S. Christensen: *Numerical Prediction of Airflow in a Room with Ceiling-Mounted Obstacles.* ISSN 0902-7513 R9256.

PAPERS ON INDOOR ENVIRONMENTAL TECHNOLOGY

PAPER NO. 32: S. G. Fox & P. V. Nielsen: *Model Experiments in 1990 and On-Site Validation in 1992 of the Air Movement in the Danish Pavilion in Seville.* ISSN 0902-7513 R9335.

PAPER NO. 33: U. Madsen, N. O. Breum & P. V. Nielsen: *Local Exhaust Ventilation - a Numerical and Experimental Study of Capture Efficiency.* ISSN 0902-7513 R9336.

PAPER NO. 34: T. V. Jacobsen, P. V. Nielsen: *Numerical Modelling of Thermal Environment in a Displacement-Ventilated Room.* ISSN 0902-7513 R9337.

PAPER NO. 35: P. Heiselberg: *Draught Risk from Cold Vertical Surfaces.* ISSN 0902-7513 R9338.

PAPER NO. 36: P. V. Nielsen: *Model Experiments for the Determination of Airflow in Large Spaces.* ISSN 0902-7513 R9339.

PAPER NO. 37: K. Svidt: *Numerical Prediction of Buoyant Air Flow in Livestock Buildings.* ISSN 0902-7513 R9351.

PAPER NO. 38: K. Svidt: *Investigation of Inlet Boundary Conditions Numerical Prediction of Air Flow in Livestock Buildings.* ISSN 0902-7513 R9407.

PAPER NO. 39: C. E. Hyldgaard: *Humans as a Source of Heat and Air Pollution.* ISSN 0902-7513 R9414.

PAPER NO. 40: H. Brohus, P. V. Nielsen: *Contaminant Distribution around Persons in Rooms Ventilated by Displacement Ventilation.* ISSN 0902-7513 R9415.

PAPER NO. 41: P. V. Nielsen: *Air Distribution in Rooms - Research and Design Methods.* ISSN 0902-7513 R9416.

PAPER NO. 42: H. Overby: *Measurement and Calculation of Vertical Temperature Gradients in Rooms with Convective Flows.* ISSN 0902-7513 R9417.

PAPER NO. 43: H. Brohus, P. V. Nielsen: *Personal Exposure in a Ventilated Room with Concentration Gradients.* ISSN 0902-7513 R9424.

PAPER NO. 44: P. Heiselberg: *Interaction between Flow Elements in Large Enclosures.* ISSN 0902-7513 R9427.

PAPER NO. 45: P. V. Nielsen: *Prospects for Computational Fluid Dynamics in Room Air Contaminant Control.* ISSN 0902-7513 R9446.

PAPER NO. 46: P. Heiselberg, H. Overby, & E. Bjørn: *The Effect of Obstacles on the Boundary Layer Flow at a Vertical Surface.* ISSN 0902-7513 R9454.

PAPER NO. 47: U. Madsen, G. Aubertin, N. O. Breum, J. R. Fontaine & P. V. Nielsen: *Tracer Gas Technique versus a Control Box Method for Estimating Direct Capture Efficiency of Exhaust Systems.* ISSN 0902-7513 R9457.

Department of Building Technology and Structural Engineering
Aalborg University, Sohngaardsholmsvej 57. DK 9000 Aalborg
Telephone: +45 98 15 85 22 Telefax: +45 98 14 82 43